

# **ELECTRONIC CHIP COOLING IN VERTICAL CONFIGURATION USING FLUENT-GAMBIT**

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE  
REQUIREMENTS FOR THE DEGREE OF

**Bachelor of Technology**

**in**

**Mechanical Engineering**

**By**

**PARESH RANJAN NAYAK**



**Department of Mechanical Engineering  
National Institute of Technology  
Rourkela  
2007**

# **ELECTRONIC CHIP COOLING IN VERTICAL CONFIGURATION USING FLUENT-GAMBIT**

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE  
REQUIREMENTS FOR THE DEGREE OF

**Bachelor of Technology**

in

**Mechanical Engineering**

By

**PARESH RANJAN NAYAK**

Under the guidance of

**Dr. ASHOK KUMAR SATAPATHY**



**Department of Mechanical Engineering  
National Institute of Technology  
Rourkela  
2007**



## **National Institute of Technology Rourkela**

### **CERTIFICATE**

This is to certify that the thesis entitled “**ELECTRONIC CHIP COOLING IN VERTICAL CONFIGURATION USING FLUENT GAMBIT**” submitted by Sri Paresh Ranjan Nayak in partial fulfillment of the requirements for the award of Bachelor of technology Degree in Mechanical Engineering at the National Institute of Technology, Rourkela (Deemed University) is an authentic work carried out by him under my supervision and guidance.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other University / Institute for the award of any Degree or Diploma.

Date

Dr. Ashok Kumar Satapathy.  
National Institute of Technology  
Rourkela-769008



## ***ACKNOWLEDGEMENT***

We deem it a privilege to have been the student of Mechanical Engineering stream in National Institute of Technology, ROURKELA

Our heartfelt thanks to Dr.A.K.Satapathy, our project guide who helped us to bring out this project in good manner with his precious suggestion and rich experience.

We take this opportunity to express our sincere thanks to our project guide for co-operation in accomplishing this project a satisfactory conclusion.

Paresh Ranjan Nayak  
Roll No. 10303045  
Department of Mechanical Engg  
National Institute of technology

# CONTENTS

<b>Contents.....</b>	<b>i</b>
<b>Abstract.....</b>	<b>ii</b>
<b>List of Figures.....</b>	<b>iii</b>
<b>1. Introduction</b>	
1.1 The miniaturization.....	2
1.2 CFD process.....	2
1.3 Cooling methods.....	3
1.4 Thermal options for different packages.....	5
<b>2. Theory</b>	
2.1 Convection.....	8
2.2 Reynolds Number.....	8
2.3 Nusselt Number.....	10
2.4 Prandtl Number.....	11
2.5 Grashof Number.....	12
2.6 Naviers-stoke equation.....	13
2.7 Discretization methods.....	14
<b>3. About the project</b>	
3.1 How does a CFD code work?.....	17
<b>4. Fluent-Gambit analysis</b>	
4.1 Design of 3-d channel using modeling tool gambit.....	20
4.2 Fluent analysis.....	22
<b>5. Results and Discussions.....</b>	<b>28</b>
<b>6. Application of CFD.....</b>	<b>39</b>
<b>7. Conclusion.....</b>	<b>40</b>
<b>Bibliography.....</b>	<b>41</b>

## **ABSTRACT**

In electronic equipments, thermal management is indispensable for its longevity and hence it is one of the important topics of current research. These electronic equipments are virtually synonymous with modern life, for instance appliances, instruments and computer specifications. The dissipation of heat is necessary for its proper function. The heat is generated by the resistance encountered by electric current. This has been further hastened by the continued miniaturization of electronic systems which causes increase in the amount of heat generation per unit volume by many folds. Unless proper cooling arrangement is designed, the operating temperature exceeds permissible limit. As a consequence, chances of failure get increased. A typical electronic system consists of several wire boards, known as printed circuit board (PCB), on which large numbers of components are mounted. These PCBs are housed in an enclosure to protect them from detrimental affect of environment, and also to protect users from electronic hazards.

The enclosure has large number of vents on it to facilitate passage of cooling air. The flow of air over the components is maintained either by fan or by free convection generated due to heated components. The rate of the cooling of components strongly depends on the shape and size of the enclosure, and also on the shape, size and location of vents. The objective of the present study is to investigate the influences of these parameters on cooling of the components.

In this project, the heat and fluid characteristics of air in a vertical channel with multiple square obstructions have been considered. The problem will be solved by using the software package FLUENT – GAMBIT. The stream lines and will be plotted for visualization and to study the heat transfer phenomena.

FLUENT is a computational fluid dynamics (CFD) software package to simulate fluid flow problems. It uses the finite-volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, etc. Geometry and grid generation is done using GAMBIT which is the preprocessor bundled with FLUENT.

## LIST OF FIGURES

<b>Sl. No.</b>	<b>Topic</b>	<b>Page</b>
<b>1</b>	Grid display showing pressure outlet, velocity inlet symmetry and chip blocks	28
<b>2</b>	Grid display showing isothermal wall, pressure outlet and velocity inlet	28
<b>3</b>	Graph shows the iteration of various residuals and convergence	29
<b>4</b>	Grid display showing the velocity inlet and three surface planes	29
<b>5</b>	Contours showing the pressure variation along XY and XZ plane	31
<b>6</b>	Contours showing the pressure variation along the chips and velocity inlet	31
<b>7</b>	Contours showing the temperature variation along YZ plane, chip and pressure outlet	32
<b>8</b>	Contour showing the temperature variation along symmetry, chip walls and velocity inlet	32
<b>9</b>	Contour showing the velocity variation along symmetry, plane-XZ and velocity inlet	33
<b>10</b>	Contour showing the velocity variation along plane-YZ ,chip walls and pressure outlet	33
<b>11</b>	Contours showing the Surface Nusselt Number variation along chip walls, velocity inlet and pressure outlet	34
<b>12</b>	Contours showing the Surface Nusselt Number variation along isothermal walls	34
<b>13</b>	Graphs showing the Surface Nusselt Number variation along symmetry	35
<b>14</b>	Graphs showing the Surface Nusselt Number variation along XY plane	35

# Chapter 1

## INTRODUCTION



# **INTRODUCTION**

## **1.1 THE MINIATURIZATION**

From 1940, since the first electronic computers and devices were discovered, the technology has come a long way. Faster and smaller computers have led to the development of faster, denser and smaller circuit technologies which further has led to increased heat fluxes generating at the chip and the package level. Over the years, significant advances have been made in the application of air cooling techniques to manage increased heat fluxes. Air cooling continues to be the most widely used method of cooling electronic components because this method is easy to incorporate and is cheaply available. Although significant heat fluxes can be accommodated with the use of liquid cooling, its use is still limited in most extreme cases where there is no choice available.

FLUENT is a computational fluid dynamics (CFD) software package to simulate fluid flow problems. It uses the finite-volume method to solve the governing equations for a fluid. It provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, etc. Geometry and grid generation is done using GAMBIT which is the preprocessor bundled with FLUENT.

## **1.2 CFD PROCESS:**

Preprocessing is the first step in building and analyzing a flow model. It includes building the model (or importing from a CAD package), applying the mesh, and entering the data. We used Gambit as the preprocessing tool in our project.

There are four general purpose products: FLUENT, Flowizard, FIDAF, and POLYFLOW. FLUENT is used in most industries All Fluent software includes full post processing capabilities.

### **1.2.1 GAMBIT CFD PREPROCESSOR:**

Fast geometry modeling and high quality meshing are crucial to successful use of CFD.GAMBIT gives us both. Explore the advantage:

Ease of use

CAD/CAE Integration

Fast Modeling

CAD Cleanup

Intelligent Meshing

EASE-OF-USE

GAMBIT has a single interface for geometry creation and meshing that brings together all of Fluent's preprocessing technologies in one environment. Advanced tools for journaling let us edit and conveniently replay model building sessions for parametric studies.

### **1.3 COOLING METHODS:**

Various cooling methods are available for keeping electronic devices within their operating temperature specifications.

#### **1.3.2 VENTING**

Natural air currents flow within any enclosure. Taking advantage of this current saves on a long term component cost. Using a computer modeling package, a designer can experiment with component placement and the addition of enclosure venting to determine an optimum solution. When this solution fails to cool the device sufficiently, the addition of a fan is often the next step.

#### **1.3.3 ENCLOSURE FANS**

The increased cooling provided by adding a fan to a system makes it a popular part of many thermal solutions. Increased air flow also Increases the cooling efficiency of heat sinks, allowing a smaller or less efficient heat sink to perform adequately.

The decision to add a fan to a system depends on number considerations. Mechanical

operation makes fans inherently less reliable than a passive system. In small enclosures, the pressure drop between the inside and the outside of the enclosure can limit the efficiency of the fan.

#### **1.3.4 PASSIVE HEAT SINKS:**

Passive heat sinks use a mass of thermally conductive material to move heat away from the device into the air stream, where it can be carried away. Heat sink designs include fins or other protrusions to increase the surface area, thus increasing its ability to remove heat from the device.

#### **1.3.5 ACTIVE HEAT SINKS:**

When a passive heat sink cannot remove heat fast enough, a small fan may be added directly to the heat sink itself, making the heat sink an active component. These active heat sinks, often used to cool microprocessors, provide a dedicated air stream for a critical device. Active heat sinks often are a good choice when an enclosure fan is impractical.

#### **1.3.6 HEAT PIPES:**

Heat pipes, a type of phase-change recirculating system, use the cooling power of vaporization to move heat from one place to another. Within a closed heat removal system, such as a sealed copper pipe, a fluid at the hot end (near a device) is changed into a vapor. Then the gas passes through a heat removal area, typically a heat sink using either air cooling or liquid cooling techniques. The temperature reduction causes the fluid to recondense into a liquid, giving off its heat to the environment. A heat pipe is a cost effective solution, and it spreads the heat uniformly throughout the heat sink condenser section, increasing its thermal effectiveness.

#### **1.3.7 METAL BACKPLANES:**

Metal-core printed circuit boards, stamped plates on the underside, of a laptop keyboard, and large copper pads on the surface of a printed circuit board all employ large metallic areas to dissipate heat.

### **1.3.8 THERMAL INTERFACES:**

The interface between the device and the thermal product used to cool it is an important factor in the thermal solution. For example, a heat sink attached to a plastic package using double sided tape cannot dissipate the same amount of heat as the same heat sink directly in contact with thermal transfer plate on a similar package.

Microscopic air gaps between a semiconductor package and the heat sink, caused by surface non-uniformity, can degrade thermal performance. This degradation increases at higher operating temperature. Interface materials appropriate to the package type reduce the variability induced by varying surface roughness.

Since the interface thermal resistance is dependent upon applied force, the contact pressure becomes an integral design parameter of the thermal solution. If a package/ device can withstand a limited amount of contact pressure, it is important that thermal calculations use the appropriate thermal resistance for that pressure.

The chemical compatibility of the interface materials with the package type is another important factor.

## **1.4 THERMAL OPTIONS FOR DIFFERENT PACKAGES:**

Many applications have different constraints that favor one thermal solution over another. Power devices need to dissipate large amount of heat. The thermal solution for microprocessors must take space constraints into account. Surface mount and ball grid array technologies have assembly considerations. Notebook computers require efficiency in every area, including space, weight, and energy usage. While the optimum solution for any one of these package types must be determined on a case-by case basis, some solutions address specific issues, making them more suitable for a particular application.

### **1.4.1 POWER DEVICES:**

Newer power devices incorporate surface mount compatibility into the power-hungry design. These devices incorporate a heat transfer plate on the bottom of the device, which can be wave soldered directly to the printed circuit board.

Metal-core substrates offer a potential solution to power device cooling, provided there are no other heat-sensitive devices in the assembly, and the cost of the board can be justified.

### **1.4.2 MICROPROCESSORS:**

As microprocessor technology advances, the system designer struggles to keep ahead of the increase in the thermal output of both (the voltage regulator and the microprocessor). The use of active heat sinks allows concentrated, dedicated cooling of the microprocessor, without severely impacting space requirements. For some applications, specially designed passive heat sinks facilitate the use of higher-powered voltage regulators in the same footprint, eliminating the need for board redesign.

### **1.4.3 BGAs:**

While BGA-packaged devices transfer more heat to the board than leaded devices, the type of package can affect the ability to dissipate sufficient heat to maintain high device reliability.

All plastic packages insulate the top of the device making heat dissipation through top mounted heat sinks difficult and more expensive. Metal heat spreaders incorporated into the top of the package enhance the ability to dissipate power from the chip

For some lower power devices flexible copper spreaders attach to pre-applied double sided tape, offering a “*quick-fix*” for border line applications

As the need to dissipate more power increases, the optimum heat sink becomes heavier. To prevent premature failure caused by ball shear, well designed of the self heat sinks include spring loaded pins or clips that allow the weight of the heat sinks to be borne by the PC-board instead of the device.

# Chapter 2

THEORY

# THEORY

## 2.1 CONVECTION

Convection in the most general terms refers to the internal movement of currents within fluids (i.e. liquids and gases). It cannot occur in solids due to the atoms not being able to flow freely.

Convection may cause a related phenomenon called advection, in which either mass or heat is transported by the currents or motion in the fluid. A common use of the term *convection* relates to the special case in which advected (carried) substance is **heat**. In this case, the heat itself may be an indirect cause of the fluid motion even while being transported by it. In this case, the problem of heat transport (and related transport of other substances in the fluid due to it) may become especially complicated.

Convection is of two types:-

1. Forced convection
2. Free Convection

**Forced Convection:** When the density difference is created by some means like blower or compressor and due to which circulation takes place then it is known as forced convection

**Free Convection:** Density variation happens naturally then it is called free convection

Here we are concerned for Newtonian fluid only (fluid that follows Newton law of cooling)

$$\tau = \mu \frac{\partial u}{\partial y}$$

## 2.2 REYNOLDS NUMBER

In fluid mechanics, the Reynolds number is the ratio of inertial forces ( $v_s \rho$ ) to viscous forces ( $\mu/L$ ) and consequently it quantifies the relative importance of these two types of forces for given flow conditions. Thus, it is used to identify different flow regimes, such as laminar or turbulent flow.

It is one of the most important dimensionless numbers in fluid dynamics and is used, usually along with other dimensionless numbers, to provide a criterion for determining dynamic similitude. When two geometrically similar flow patterns, in perhaps different fluids with possibly different flow-rates, have the same values for the relevant dimensionless numbers, they are said to be dynamically similar.

It is named after Osborne Reynolds (1842–1912), who proposed it in 1883. Typically it is given as follows:

$$Re = \frac{\rho v_s L}{\mu} = \frac{v_s L}{\nu} = \frac{\text{Inertial forces}}{\text{Viscous forces}}$$

Where:

$v_s$  - Mean fluid velocity,

$L$  - Characteristic length,

$\mu$  - (Absolute) dynamic fluid viscosity,

$\nu$  - Kinematic fluid viscosity:  $\nu = \mu / \rho$ ,

$\rho$  - Fluid density.

For flow in pipes for instance, the characteristic length is the pipe diameter, if the cross section is circular, or the hydraulic diameter, for a non-circular cross section.

Laminar flow occurs at low Reynolds numbers, where viscous forces are dominant, and is characterized by smooth, constant fluid motion, while turbulent flow, on the other hand, occurs at high Reynolds numbers and is dominated by inertial forces, producing random eddies, vortices and other flow fluctuations.

The transition between laminar and turbulent flow is often indicated by a critical Reynolds number ( $Re_{crit}$ ), which depends on the exact flow configuration and must be determined experimentally. Within a certain range around this point there is a region of gradual transition where the flow is neither fully laminar nor fully turbulent, and predictions of fluid behaviour can be difficult. For example, within circular pipes the critical Reynolds number is generally accepted to be 2300, where the Reynolds number is based on the pipe diameter and the mean velocity  $v_s$  within the pipe, but engineers will avoid any pipe configuration that falls within the range of Reynolds numbers from about 2000 to 3000 to ensure that the flow is either laminar or turbulent.



For flow over a flat plate, the characteristic length is the length of the plate and the characteristic velocity is the free stream velocity. In a boundary layer over a flat plate the local regime of the flow is determined by the Reynolds number based on the distance measured from the leading edge of the plate. In this case, the transition to turbulent flow occurs at a Reynolds number of the order of  $10^5$  or  $10^6$ .

## **2.3 NUSSELT NUMBER**

The Nusselt number is a dimensionless number that measures the enhancement of heat transfer from a surface that occurs in a 'real' situation, compared to the heat transferred if just conduction occurred. Typically it is used to measure the enhancement of heat transfer when convection takes place.

$$Nu_L = \frac{hL}{k_f} = \frac{\text{Convective heat transfer}}{\text{Conductive heat transfer}}$$

Where

$L$  = characteristic length, which is simply Volume of the body divided by the Area of the body (useful for more complex shapes)

$k_f$  = thermal conductivity of the "fluid"

$h$  = convection heat transfer coefficient

Selection of the significant length scale should be in the direction of growth of the boundary layer. A salient example in introductory engineering study of heat transfer would be that of a horizontal cylinder versus a vertical cylinder in free convection.

Several empirical correlations are available that are expressed in terms of Nusselt number in the elementary analysis of flow over a flat plate etc. Sieder-Tate, Colburn and many others have provided such correlations.

For a local Nusselt number, one may evaluate the significant length scale at the point of interest. To obtain an average Nusselt number analytically one must integrate over the characteristic length. More commonly the average Nusselt number is obtained by the pertinent correlation equation, often of the form  $Nu = Nu(Ra, Pr)$ .

The Nusselt number can also be viewed as being a dimensionless temperature gradient at the surface.

## **2.4 PRANDTL NUMBER**

The Prandtl number is a dimensionless number approximating the ratio of momentum diffusivity (viscosity) and thermal diffusivity. It is named after Ludwig Prandtl.

It is defined as:

$$Pr = \frac{\nu}{\alpha} = \frac{\text{viscous diffusion rate}}{\text{thermal diffusion rate}}$$

Where:

$\nu$  is the kinematic viscosity,  $\nu = \mu / \rho$ .

$\alpha$  is the thermal diffusivity,  $\alpha = k / (\rho c_p)$ .

Typical values for  $Pr$  are:

- around 0.7 for air and many other gases,
- around 7 for water
- around  $7 \times 10^{21}$  for Earth's mantle
- between 100 and 40,000 for engine oil,
- between 4 and 5 for R-12 refrigerant
- around 0.015 for mercury

For mercury, heat conduction is very effective compared to convection: thermal diffusivity is dominant. For engine oil, convection is very effective in transferring energy from an area, compared to pure conduction: momentum diffusivity is dominant.

In heat transfer problems, the Prandtl number controls the relative thickness of the momentum and thermal boundary layers.

The mass transfer analog of the Prandtl number is the Schmidt number.

## 2.5 GRASHOF NUMBER

The **Grashof number** is a dimensionless number in fluid dynamics which approximates the ratio of the buoyancy force to the viscous force acting on a fluid. It is named after the German engineer Franz Grashof.

$$Gr = \frac{g\beta(T_s - T_\infty)L^3}{\nu^2}$$

Where

$g$  = acceleration due to Earth's gravity

$\beta$  = volumetric thermal expansion coefficient

$T_s$  = source temperature

$T_\infty$  = quiescent temperature

$L$  = characteristic length

$\nu$  = kinematic viscosity

The product of the Grashof number and the Prandtl number gives the Rayleigh number, a dimensionless number that characterizes convection problems in heat transfer.

There is an analogous form of the **Grashof number** used in cases of natural convection mass transfer problems.

$$Gr_c = \frac{g\beta^*(C_{a,s} - C_{a,a})L^3}{\nu^2}$$

Where

$$\beta^* = -\frac{1}{\rho} \left( \frac{\partial \rho}{\partial C_a} \right)_{T,p}$$

And

$g$  = acceleration due to Earth's gravity

$C_{a,s}$  = concentration of species  $a$  at surface

$C_{a,a}$  = concentration of species  $a$  in ambient medium

$L$  = characteristic length

$\nu$  = kinematic viscosity

$\rho$  = fluid density

$C_a$  = concentration of species  $a$

$T$  = constant temperature

$p$  = constant pressure

## 2.6 NAVIER-STOKES EQUATION

The Navier-Stokes equations are derived from conservation principles of:

- Mass
- Energy
- Momentum
- Angular momentum
- Equation of continuity

Conservation of mass is written:

$$\begin{aligned}\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) &= 0 \\ &= \frac{\partial \rho}{\partial t} + \rho \nabla \cdot \mathbf{v} + \mathbf{v} \cdot \nabla \rho \\ &= \frac{D\rho}{Dt} + \rho \nabla \cdot \mathbf{v} = 0\end{aligned}$$

Where  $\rho$  is the mass density (mass per unit volume), and  $\mathbf{v}$  is the velocity of the fluid.

In the case of an incompressible fluid,  $\rho$  does not vary along a path-line and the equation reduces to:

$$\nabla \cdot \mathbf{v} = 0$$

The Navier-Stokes Continuity equation for cylindrical coordinates is:

$$\frac{\partial u_r}{\partial r} + \frac{u_r}{r} + \frac{1}{r} \frac{\partial u_\theta}{\partial \theta} + \frac{\partial u_z}{\partial z}$$

Note that the Navier-Stokes equations can only describe fluid flow approximately and that, at very small scales or under extreme conditions, real fluids made out of mixtures of discrete molecules and other material, such as suspended particles and dissolved gases, will produce different results from the continuous and homogeneous fluids modelled by the Navier-Stokes equations.

$$\rho \left( \frac{\partial u_z}{\partial t} + u_r \frac{\partial u_z}{\partial r} + \frac{u_\theta}{r} \frac{\partial u_z}{\partial \theta} + u_z \frac{\partial u_z}{\partial z} \right) = -\frac{\partial P}{\partial z} + \mu \left( \frac{\partial^2 u_z}{\partial r^2} + \frac{1}{r} \frac{\partial u_z}{\partial r} + \frac{1}{r^2} \frac{\partial^2 u_z}{\partial \theta^2} + \frac{\partial^2 u_z}{\partial z^2} \right) + F_z$$

Note that the Navier-Stokes equations can only describe fluid flow approximately and that, at very small scales or under extreme conditions, real fluids made out of mixtures of discrete molecules and other material, such as suspended particles and dissolved gases, will produce different results from the continuous and homogeneous fluids modeled by the Navier-Stokes equations.

## **2.7 DISCRETIZATION METHODS:**

The stability of the chosen discretization is generally established numerically rather than analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier-Stokes equations both admit shocks, and contact surfaces.

Some of the discretization methods being used are:

### **2.7.1 FINITE VOLUME METHOD:**

This is the "classical" or standard approach used most often in commercial software and research codes. The governing equations are solved on discrete control volumes. This integral approach yields a method that is inherently conservative (i.e., quantities such as density remain physically meaningful):

$$\frac{\partial}{\partial t} \iiint Q dV + \iint F d\mathbf{A} = 0,$$

Where  $Q$  is the vector of conserved variables,  $F$  is the vector of fluxes (see Euler equations or Navier-Stokes equations),  $V$  is the cell volume, and  $A$  is the cell surface area.

### 2.7.2 FINITE ELEMENT METHOD:

This method is popular for structural analysis of solids, but is also applicable to fluids. The FEM formulation requires, however, special care to ensure a conservative solution. The FEM formulation has been adapted for use with the Navier-Stokes equations. In this method, a weighted residual equation is formed:

$$R_i = \iiint V^e W_i Q dV^e$$

Where  $R_i$  is the equation residual at an element vertex  $i$ ,  $Q$  is the conservation equation expressed on an element basis,  $W_i$  is the weight factor and  $V^e$  is the volume of the element.

### 2.7.3 FINITE DIFFERENCE METHOD:

This method has historical importance and is simple to program. It is currently only used in few specialized codes. Modern finite difference codes make use of an embedded boundary for handling complex geometries making these codes highly efficient and accurate. Other ways to handle geometries are using overlapping-grids, where the solution is interpolated across each grid.

$$\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial H}{\partial z} = 0$$

Where  $Q$  is the vector of conserved variables, and  $F$ ,  $G$ , and  $H$  are the fluxes in the  $x$ ,  $y$ , and  $z$  directions respectively.

**Boundary element method:** The boundary occupied by the fluid is divided into surface mesh.

High resolution schemes are used where shocks or discontinuities are present. To capture sharp changes in the solution requires the use of second or higher order numerical schemes that do not introduce spurious oscillations. This usually necessitates the application of flux limiters to ensure that the solution is total variation diminishing.

# Chapter 3

ABOUT THE PROJECT

## ABOUT THE PROJECT

In this project, the heat and fluid characteristics of air in a vertical channel with multiple square obstructions have been considered. The problem will be solved using FLUENT-GAMBIT software package which have FLUENT 6.0 and GAMBIT 2.0 bundled into one package to solve CFD problems. GAMBIT is used to create the initial geometry and meshing. FLUENT is used to analyze and solve the geometry with the boundary conditions and specifications mentioned.

Different layouts of multiple square obstructions will be considered including a parallel layout and a zigzag configuration. The heat transfer in each case will be calculated and the configuration having maximum heat transfer rate will be the optimal one and will be used for future considerations. The isotherms and streamlines will be plotted for flow visualization and to study the heat transfer phenomena for this configuration. The distribution of temperature and pressure of the fluid (air) along the chip face and wall are also analyzed.

The figure below shows two substrates having electronic chip arranged on them in a zigzag fashion. Air flows from a fan in order to cool the chips the area of the chips is assumed to be square type.

### **3.1 HOW DOES A CFD CODE WORK?**

CFD codes are structured around the numerical algorithms that can be tackle fluid problems. In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces input problem parameters and to examine the results. Hence all codes contain three main elements:

- 1. Pre-processing.*
- 2. Solver*
- 3. Post –processing.*

**3.1.1 PRE-PROCESSING:** Preprocessor consists of input of a flow problem by means of an operator friendly interface and subsequent transformation of this input into form of suitable for the use by the solver.



*The user activities at the Pre-processing stage involve:* Definition of the geometry of the region: The computational domain. Grid generation is the subdivision of the domain into a number of smaller, non-overlapping sub domains (or control volumes or elements Selection of physical or chemical phenomena that need to be modeled).

*Definition of fluid properties:* Specification of appropriate boundary conditions at cells, which coincide with or touch the boundary.

The solution of a flow problem (velocity, pressure, temperature etc.) is defined at nodes inside each cell. The accuracy of CFD solutions is governed by number of cells in the grid. In general, the larger numbers of cells better the solution accuracy. Both the accuracy of the solution & its cost in terms of necessary computer hardware & calculation time are dependent on the fineness of the grid. Efforts are underway to develop CFD codes with a (self) adaptive meshing capability. Ultimately such programs will automatically refine the grid in areas of rapid variation.

**3.1.2 SOLVER:** These are three distinct streams of numerical solutions techniques: finite difference, finite volume & finite element methods. In outline the numerical methods that form the basis of solver performs the following steps

The approximation of unknown flow variables are by means of simple functions.

Discretization by substitution of the approximation into the governing flow equations & subsequent mathematical manipulations.

**3.1.3 POST-PROCESSING:** As in the pre-processing huge amount of development work has recently has taken place in the post processing field. Owing to increased popularity of engineering work stations, many of which has outstanding graphics capabilities, the leading CFD are now equipped with versatile data visualization tools. These include

- Domain geometry & Grid display.
- Vector plots.
- Line & shaded contour plots.
- 2D & 3D surface plots.
- Particle tracking.
- View manipulation (translation, rotation, scaling etc.)

# Chapter 4

FLUENT-GAMBIT ANALYSIS

# DESIGN OF 3-D CHANNEL USING MODELLING TOOL GAMBIT

## 4.1 PROCEDURE(3-D VERSION OF GAMBIT)

### **STEP 1:**

Specify that the mesh to be created is for use with FLUENT 6.0:

**Main Menu > Solver > FLUENT 5/6**

Verify this has been done by looking in the *Transcript Window* where you should see. The boundary types that you will be able to select in the third step depends on the solver selected.

### **STEP 2:**

Select volume as tool geometry

**TOOL → GEOMETRY → VOLUME**

The initial 3-D brick of size 0.150 x 0.050 x 0.025 cubic mt .

Create chips (brick) of size 0.030 x 0.020 x 0.010 cubic mt.

Arrange them in a parallel configuration at equal distances.

**TOOL → GEOMETRY → SUBTRACT**

Chip volume is subtracted from channel volume.

Single volume is created.

### **STEP 3 : ( mesh volume )**

**MESH → VOLUME**

Element= Hex

Type= Cooper

Apply scheme

Interval Size= 0.002

#### **STEP 4 :( set boundary types)**

##### **ZONES → SPECIFY BOUNDARY TYPES.**

The Chip wall is selected under wall domain.

The three outside walls of the rectangular channel are selected as isothermal walls under wall domain

The upper face in X-Z plane was selected as under pressure outlet domain.

The inlet face was selected under velocity inlet domain in X-Z plane.

The face having chips is selected as symmetry.

##### **ZONES → SPECIFY BOUNDARY ZONES**

By default fluid zone was created on the faces and named as air

#### **STEP 5: (Export the mesh and save the session)**

##### **FILE → EXPORT → MESH**

File name was entered for the file to be exported. Accept was not clicked as it was 3-D model

Gambit session was saved and exit was clicked.

##### **FILE → EXIT**

## **4.2 ANALYSIS IN FLUENT**

### **PROCEDURE: (3D VERSION OF FLUENT)**

#### **STEP 1: (GRID)**

**FILE → READ → CASE**

The file channel mesh is selected by clicking on it under files and

Then ok is clicked.

The grid is checked.

**GRID → CHECK**

The grid was scaled to 1 in all x, y and z directions.

**GRID → SCALE**

The grid was displayed.

**DISPLAY → GRID**

Grid is copied in ms-word file.

#### **STEP 2 :( Models)**

The solver was specified.

**DEFINE → MODELS → SOLVER**

Solver is segregated

Implicit formulation

Space steady

Time steady

**DEFINE → MODEL → ENERGY**

Energy equation is clicked on.

**DEFINE → MODELS → VISCOUS**

The standard k-ε turbulence model was turned on.

k-ε model(2-equation)- Standard

Model constants

Cmu-0.09

C1-Epsilon-1.44

C2-Epsilon-1.92

Energy Prandtl number= 0.85

Wall Prandtl number= 0.85

KE Prandtl Number 1

No viscosity

### **STEP 3:(Materials)**

By default the material selected was air with properties.

Dynamic Viscosity,  $\mu = 1.7894 \times 10^{-5}$

Density,  $\rho = 1.225 \text{ kg/m}^3$ .

Thermal Conductivity,  $K = 0.0242 \text{ W/mK}$

Specific heat,  $C_p = 1.007 \text{ kJ/kg K}$

Molecular weight= 28.966

### **STEP 4(Operating conditions)**

Operating pressure= 101.325 KPa

Gravity =  $-9.81 \text{ m/s}^2$  in Y-direction

### **STEP 5:(Boundary conditions)**

**DEFINE → BOUNDARY CONDITIONS**

**AIR**

Set option is clicked

**Default Interior**

Set option is clicked

**Velocity inlet.**

Y -component of velocity=  $2 \text{ m/s}$ .

**Pressure outlet.**

Gauge pressure =0 Pascal.

Back flow temperature=305K.

### **Wall 1 (Isothermal Wall)-Channel Wall**

Temperature= 350K

Wall thickness=0mm.

Material= Aluminium

### **Wall 2 (Constant heat flux generation)-Chip Wall**

Constant Heat Flux,  $W = 1000\text{W/m}^2$

Wall thickness=0mm.

Material= Aluminium

### **STEP 6: (Solution)**

#### **SOLVE → CONTROLS→SOLUTIONS**

All flow, turbulent and energy equation used.

Under relaxation factors

Pressure= 0.3

Density= 1

Body Force= 1

Momentum= 0.08

#### **SOLVE →INITIALIZE**

Compute from velocity inlet= 2m/s

Click **INIT**

~Velocity inlet was chosen from the computer from the list.

~ X component of velocity is 0.

~ Init was clicked and panel was closed.

The plotting of residuals was enabled for the calculation.

#### **PLOT→ RESIDUALS**

Plot option was clicked than ok was clicked.

The case file was created

#### **FILE→WRITE-→CASE**

The calculation was started by requesting 1000 iterations

#### **SOLVE→ITERATE**

Input 100 as the number of iterations and iterate was clicked.

Convergence was checked.

Converged in 52 iterations

### **REPORT→FLUXES**

Mass flow rate

Total heat transfer

Radiation heat transfer

Their values along all zones are computed

The data was saved.

### **FILE→WRITE→DATA**

Arbitrary planes were created along XYZ Cartesian coordinate system for contour evaluation for flow inside the channel.

### **STEP 6:(Displaying the preliminary solution)**

#### ***Display of filled contours of velocity magnitude***

#### **DISPLAY→CONTOURS**

~ Velocity was selected and then velocity magnitude in the

Drop down list was selected.

~filled under option was selected.

~Display was clicked.

#### ***Display of filled contours of temperature***

#### **DISPLAY→CONTOURS**

~Temperature was selected and then

1. Static temperature.
2. Total temperature from drop down list was selected

~Display was clicked.

#### ***Display of filled contours of turbulence***

#### **DISPLAY→CONTOURS**

~ Turbulence was selected along various planes

~filled under option was selected.

~Display was clicked.

#### ***Display of filled contours of wall fluxes***

#### **DISPLAY→CONTOURS**



~ Wall fluxes was selected and then surface nusselt number in the Drop down list was selected.  
 ~filled under option was selected.  
 ~Display was clicked.

### ***Display velocity vector.***

#### **DISPLAY→VECTOR**

~Display was clicked to plot the velocity vectors.

An XY plot of temperature across the exit was created.

#### **PLOT→XY PLOT**

~ Wall fluxes were selected and Surface Nusselt in the drop down list under the Y axis functions.

The write option is clicked to save data in ms-excel file. It contains surface Nusselt number values with there node points.

The excel file is opened and we plot a graph of surface Nusselt number along Y-axis and position on Y-axis in grid display in meter along X-axis.

The graph we plot is along XZ plane and symmetry

~ Plot was clicked.

~ The average value of surface Nusselt along these two planes is calculated as 189 and 210  
 Similarly, We write the data of surface Nusselt along all node points including all zones in ms-excel file and calculate average surface Nusselt number. The value was found out to be 214

### **CALCULATION OF REYNOLDS NUMBER**

$$R = \rho v d / \mu$$

$$= (1.225 \times 2 \times 0.035) / (1.7894 \times 10^{-5})$$

$$= 4792.11 \text{ [Hence the flow is turbulent (Rectangular channel flow)]}$$

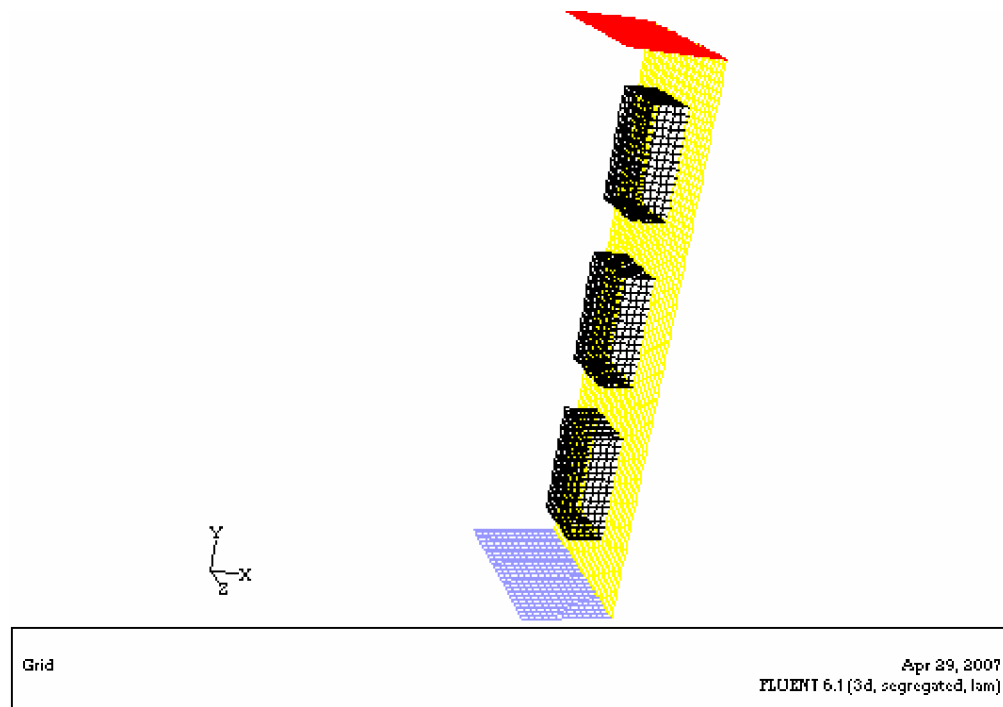
After doing many simulations with varying mesh element, type and spacing between the grids the values of surface Nusselt number was found to be same on average hence through grid independence check our simulation values are found to be correct.

The graphs and contours related to CFD analysis done above is shown from next page.

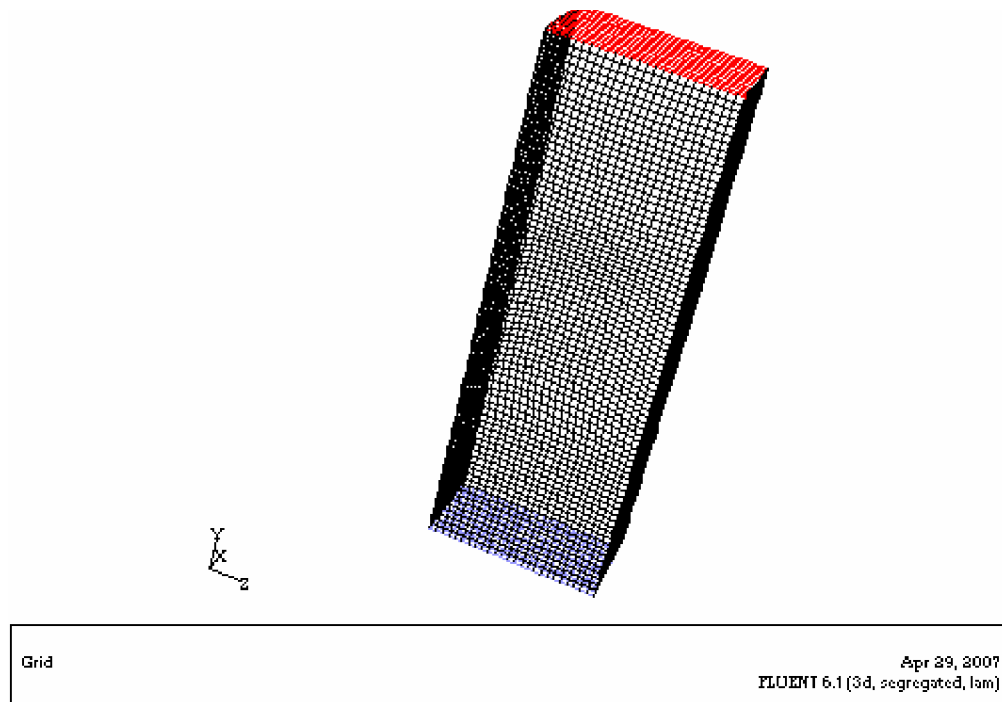
# Chapter 5

DISCUSSION

## GRID DISPLAY

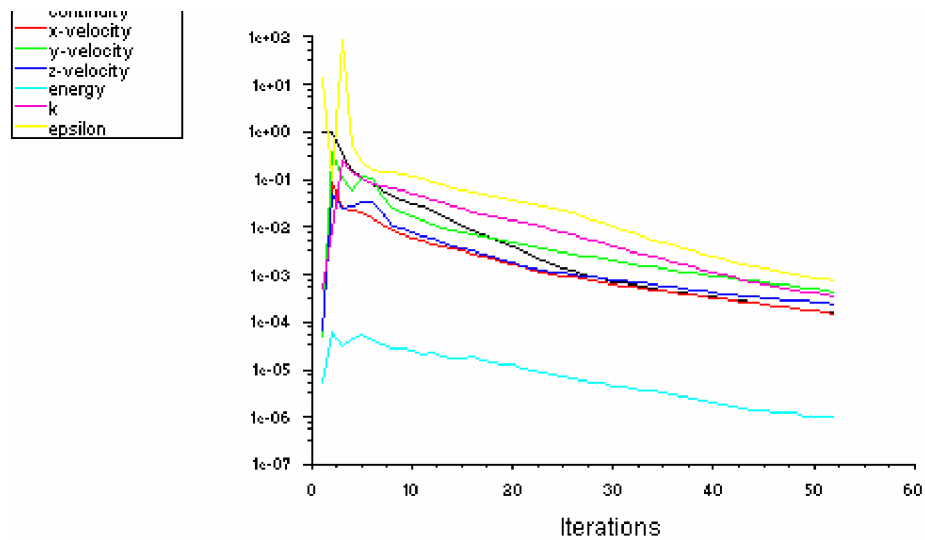


**Fig 5.1 Grid display showing pressure outlet, velocity inlet symmetry and chip blocks**



## **ITERATION GRAPH AND GRID**

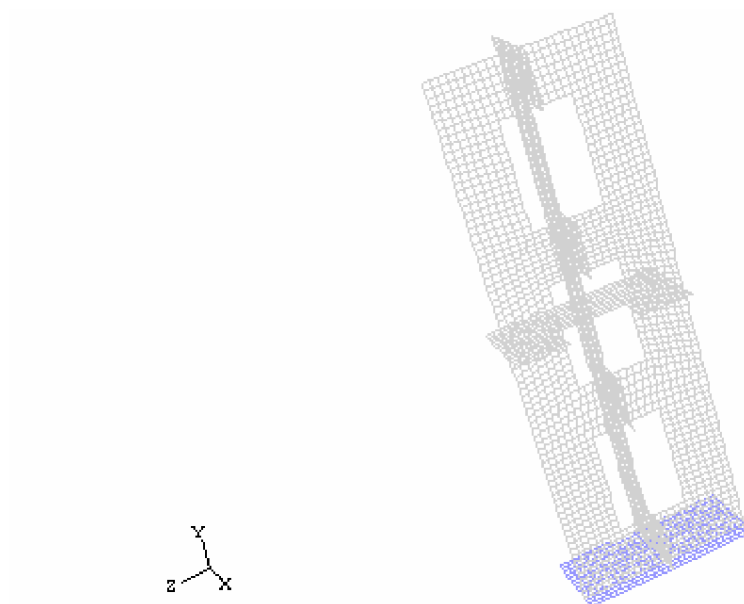
**Fig 5.2 Grid display showing boundary wall, pressure outlet and velocity inlet**



Scaled Residuals

Apr 29, 2007  
FLUENT 6.1 (3d, segregated, ske)

**Fig 5.3 Graph shows the iteration of various residuals and convergence**



Grid

Apr 29, 2007  
FLUENT 6.1 (3d, segregated, ske)

**Fig 5.4 Grid display showing the velocity inlet and three surface planes**

## **DESCRIPTION**

### **GRID DISPLAY**

#### **FIGURE 5.1**

The figure shows the partial grid pattern of 3-D rectangular channel. The grid pattern is created with Gambit software. The isometric view shows velocity inlet, chip walls, symmetry and pressure outlet respectively by blue color, black color, yellow color and red color. The fluid flow inside default interior space consists of air. The air enters from velocity inlet face and exits from pressure outlet face. The air flow inside channel cool chip walls generating constant heat flux of  $1000 \text{ W/m}^2$ .

#### **FIGURE 5.2**

The figure shows another isometric view of grid, consisting of isothermal walls, velocity inlet and pressure outlet represented by black color, blue color and red color. The isothermal walls have the constant temperature of 350K. The XYZ Cartesian coordinate system is shown below the figure.

### **ITERATION AND SURFACE PLANES**

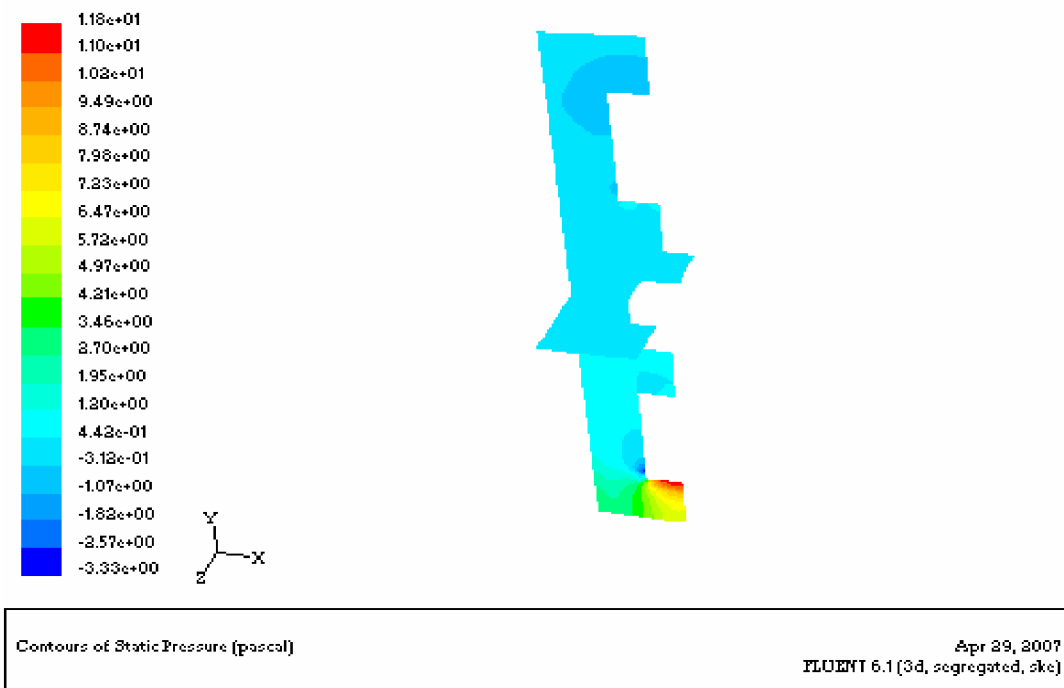
#### **FIGURE 5.3**

The figure shows the convergence iteration graph. The iteration graph has convergence of various scaled residuals consisting of continuity, x velocity, y velocity, z velocity, energy, K and epsilon. The solution has been converged in 52 iterations.

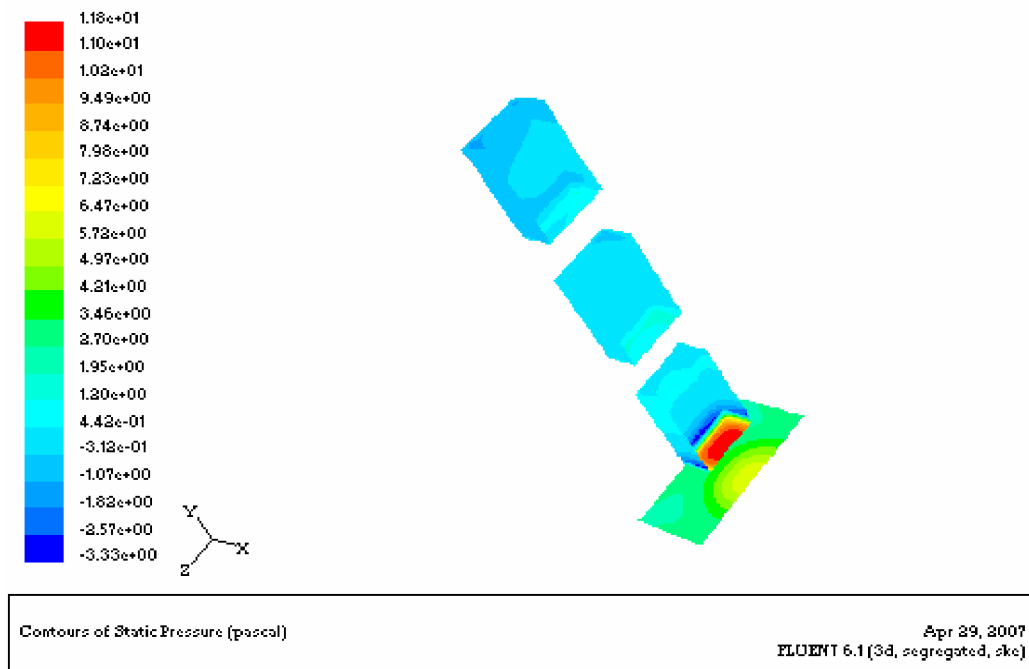
#### **FIGURE 5.4**

The figure shows various surface planes inside the channel geometry. There are three planes, plane-XY, plane-YZ, plane XZ shown by black color. The figure also shows the velocity inlet (blue color) at the bottom from where the air enters. These surface planes help in understanding the various contours and graphs of pressure velocity and temperature.

## CONTOURS OF PRESSURE

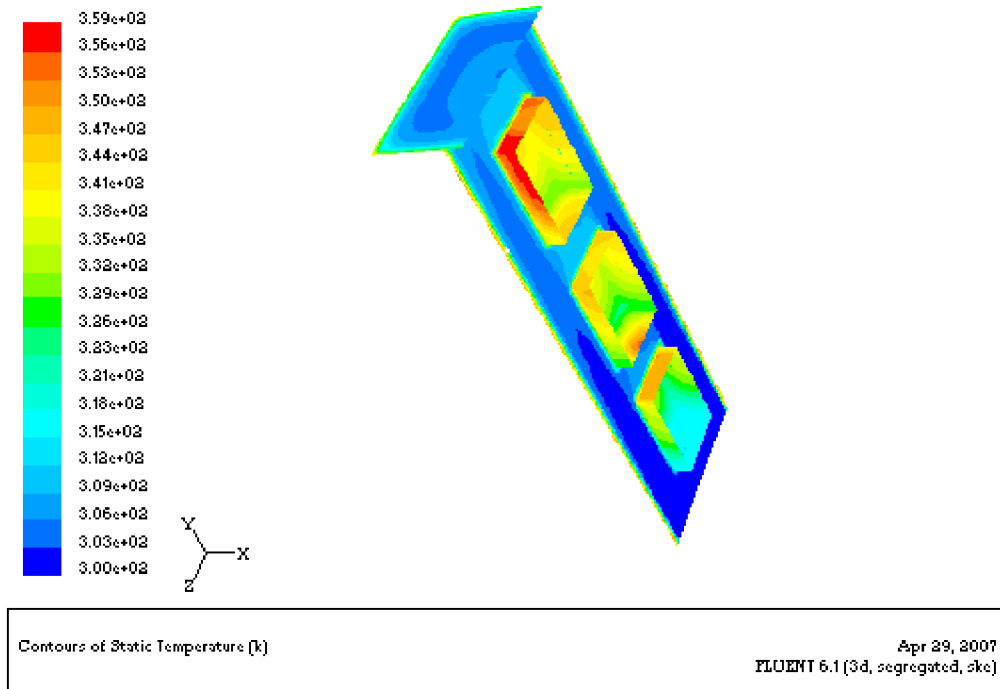


**Fig 5.5: Contours showing the pressure variation along XY and XZ plane**

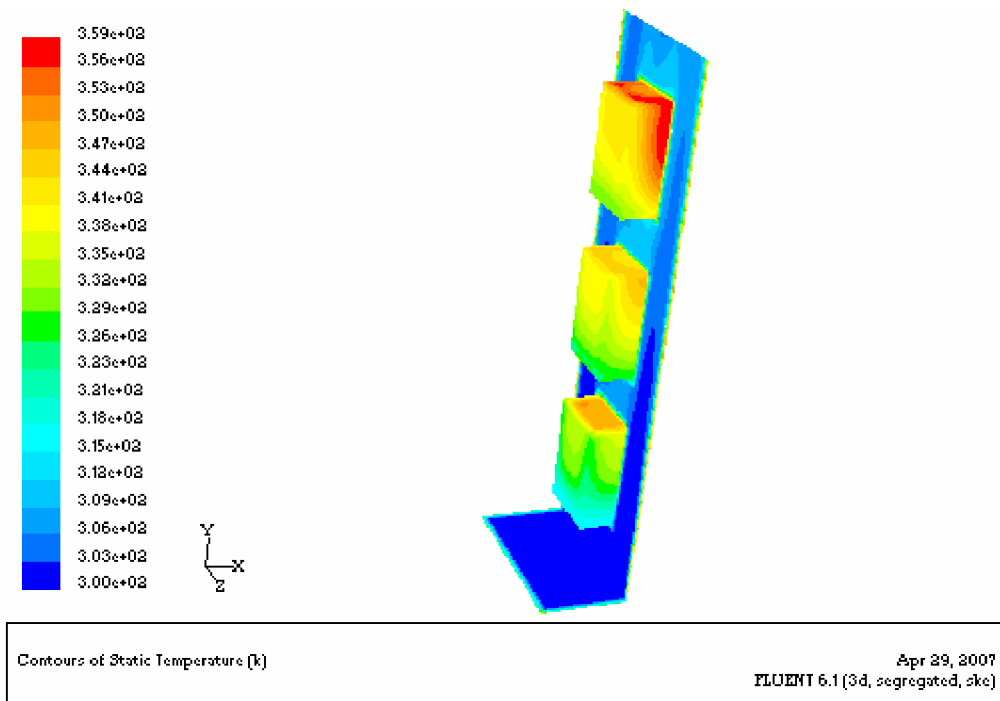


**Fig 5.6: Contours showing the pressure variation along the chips and velocity inlet**

## CONTOURS OF TEMPERATURE

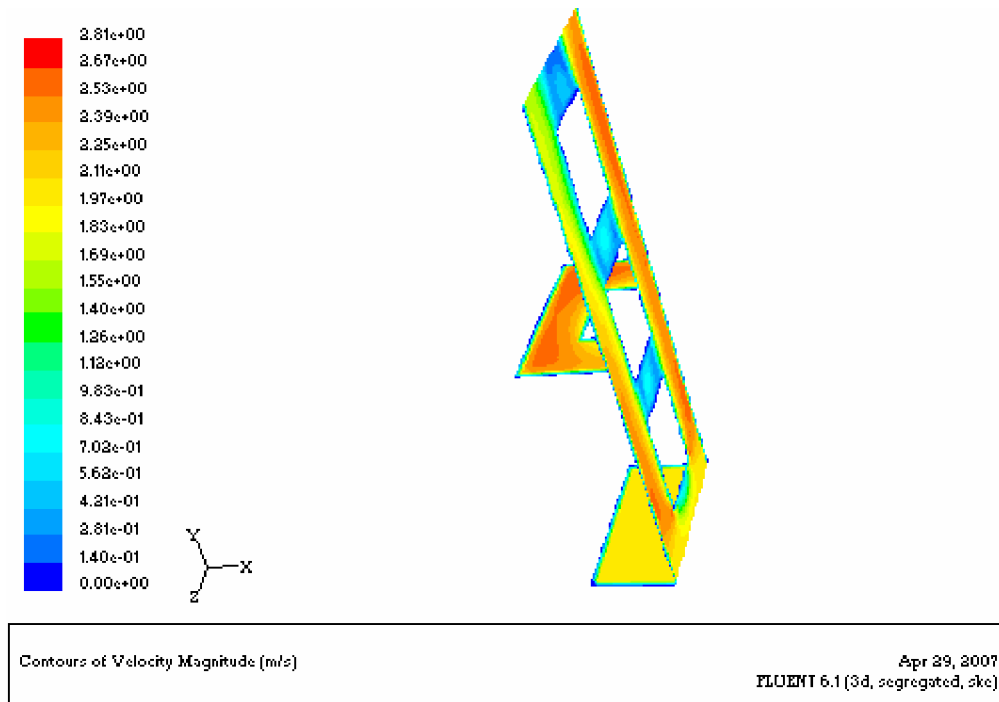


**Fig 5.7: Contours showing the temperature variation along YZ plane, chip walls and pressure outlet**

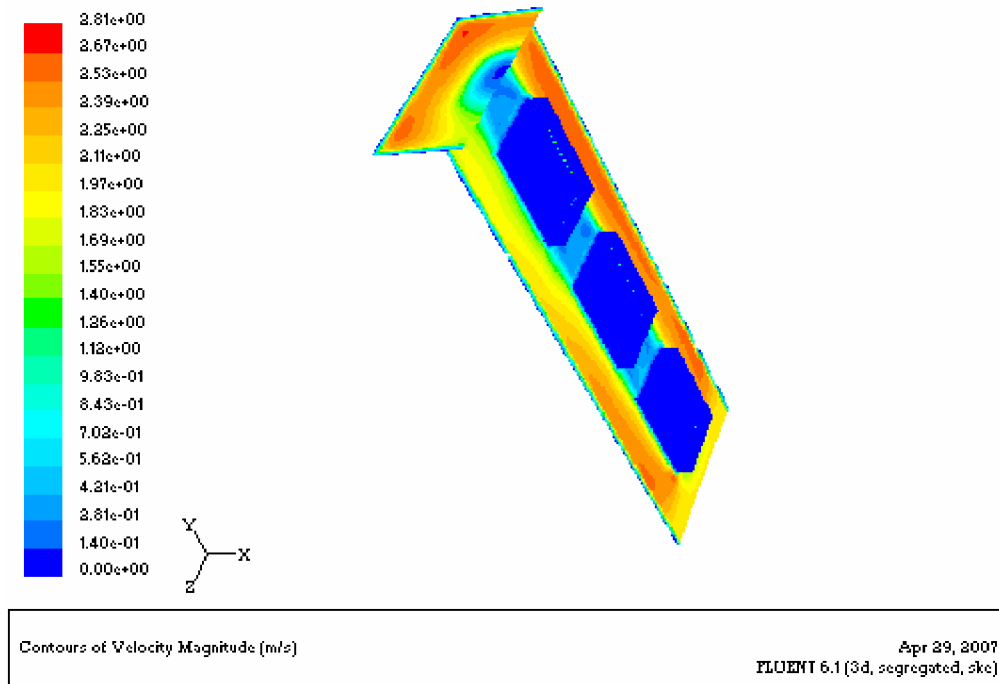


**Fig 5.8: Contour showing the temperature variation along symmetry, chip walls and velocity inlet**

## CONTOURS OF VELOCITY



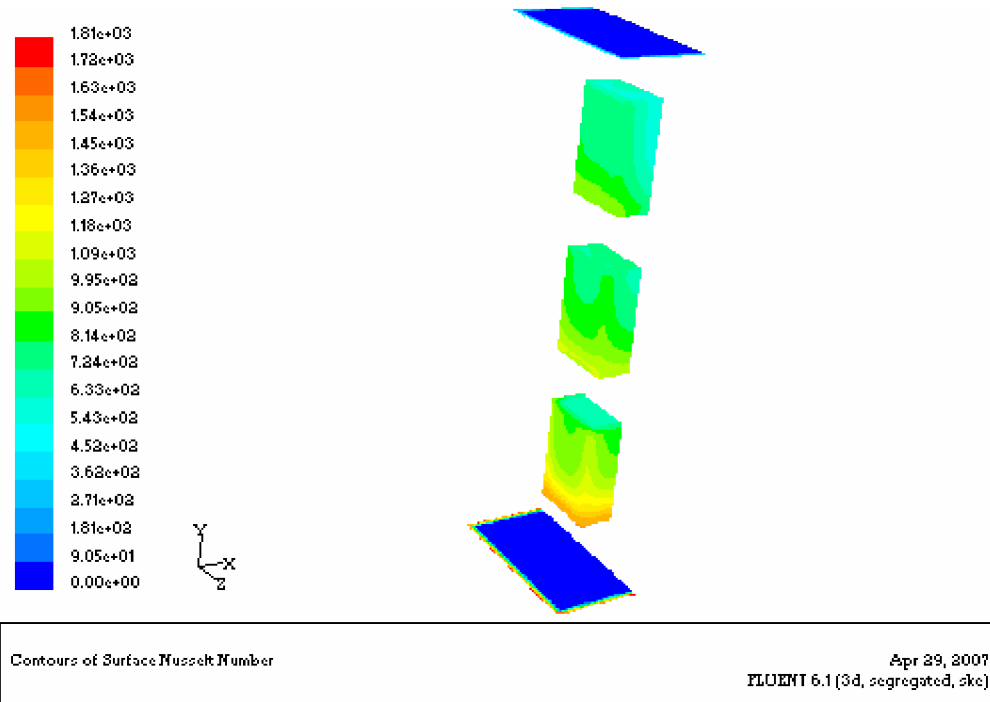
**Fig 5.9: Contour showing the velocity variation along symmetry, plane-XZ and velocity inlet**



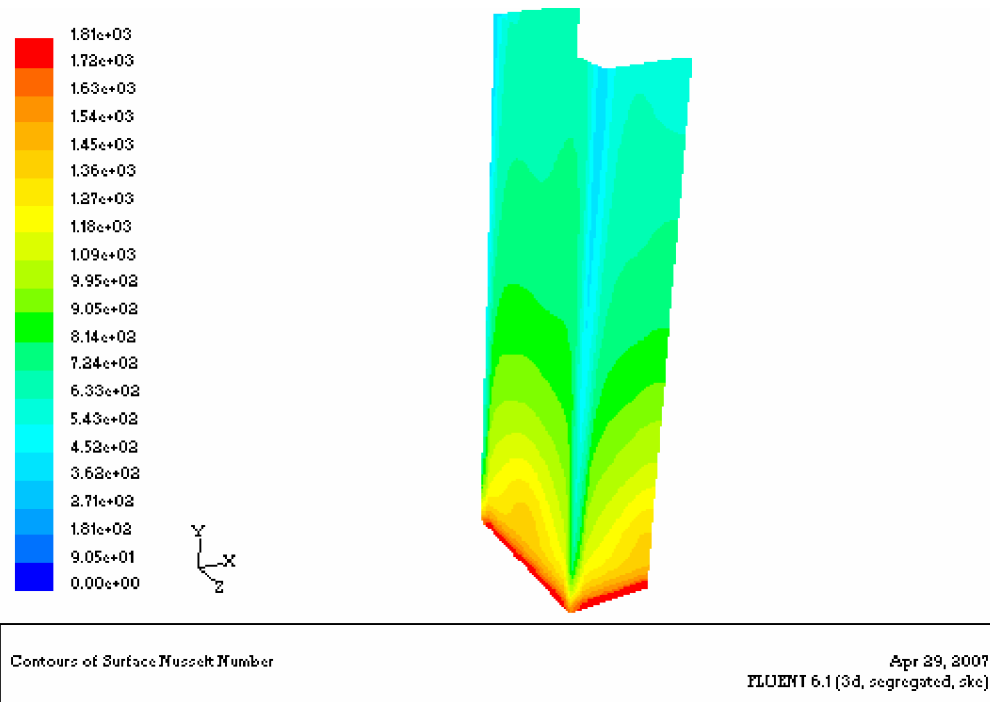
**Fig 5.10: Contour showing the velocity variation along plane-YZ ,chip walls and pressure outlet**



## CONTOURS OF SURFACE NUSSELT NUMBER

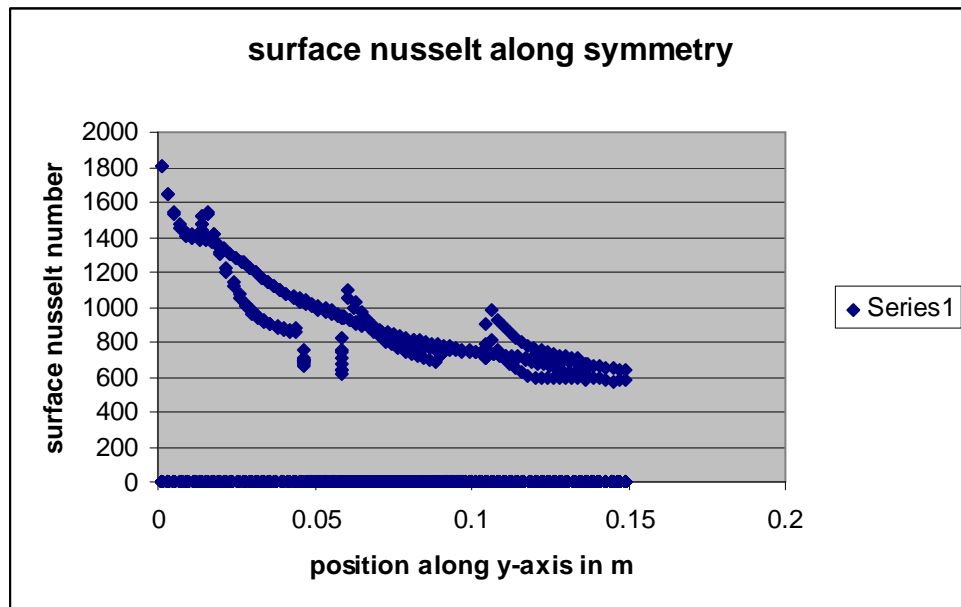


**Fig 5.11: Contours showing the Surface Nusselt Number variation along chip walls, velocity inlet and pressure outlet**

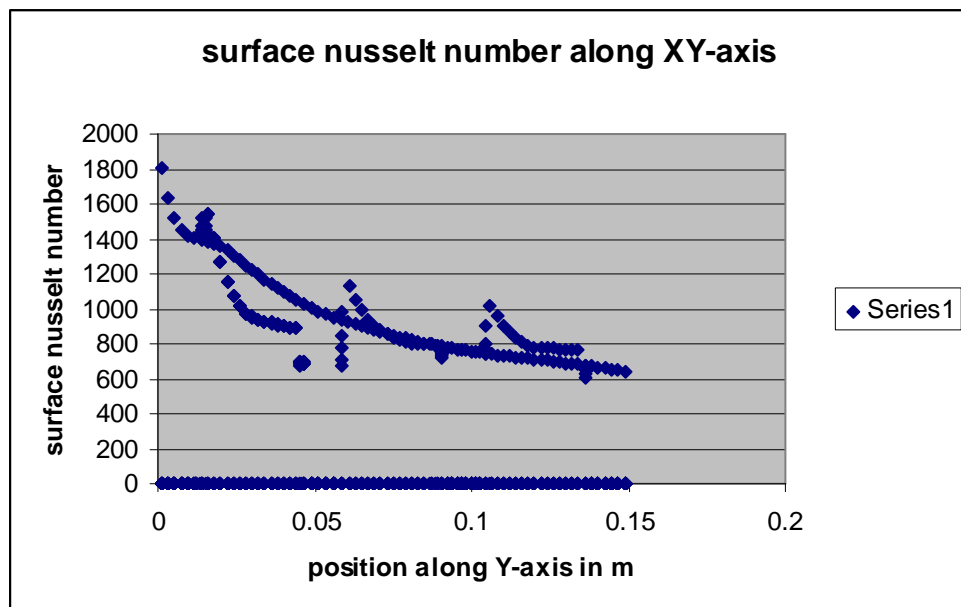


**Fig 5.12: Contours showing the Surface Nusselt Number variation along isothermal walls**

## GRAPHS OF SURFACE NUSSOLT NUMBER



**Fig 5.13 Graphs showing the Surface Nusselt Number variation along symmetry**



**Fig 5.14 Graphs showing the Surface Nusselt Number variation along XY plane**

## **CONTOURS OF PRESSURE**

### **FIGURE 5.5**

The figure shows contours of pressure variation along plane-XY and plane-XZ. In XY-plane the pressure is maximum at the front of first chip shown by red color. It slowly decreases from as we move up from reddish to greenish. At first chip there is bluish tinge at top frontal position showing pressure is minimum there. The pressure variation is minimum at backside of last chip shown by dark bluish region. In plane-XZ there is no variation of pressure at the mid of channel shown by blue color.

### **FIGURE 5.6**

The figure shows contours of pressure variation along chip walls and velocity inlet. The pressure variation along the three chips is represented by change in color. The pressure value is decreased along Y-axis direction. It is found that pressure at velocity inlet face has some value shown by greenish color. The velocity decreases to minimum (blue) at pressure outlet face.

## **CONTOURS OF TEMPERATURE**

### **FIGURE 5.7**

The figure shows contours of temperature variation along plane YZ, chip walls and pressure outlet face. Due to cooling effect of inlet air from velocity inlet face, the temperature of the chips increases as we move from bottom to the top. This can be represented by red color at the back face of last chip and bluish color at the front of the first chip. Along YZ plane temperature is lower near velocity inlet face (dark bluish color) and the color fades to light blue color as we move above.

### **FIGURE 5.8**

The figure shows contours of temperature variation along symmetry chip wall and velocity inlet face. The temperature along the edges of velocity inlet face is constant. The temperature variation along the symmetry shows that temperature is lower near velocity inlet face (dark bluish color) and the color fades to light blue color as we move above.

## **CONTOURS OF VELOCITY**

### **FIGURE 5.9**

The figure shows contours of velocity variation along symmetry, velocity inlet face and XZ-plane. At velocity inlet face, the velocity is zero just along the isothermal walls and is constant 2m/s in the default interior area. As fluid air moves along the Y-axis inside the channel, the velocity increases in the region just above the chip shown by the reddish color due to decrease in cross-sectional area. The velocity just behind the chip is zero and shown by the bluish color

### **FIGURE 5.10**

The figure shows contours of velocity variation along YZ-plane, chip walls and pressure outlet. The velocity is zero at surface of chip wall shown by blue color. At the pressure outlet the velocity is minimum along the line of chip and velocity is maximum along the regions around the chips. The variation of velocity along the YZ-plane is similar to the velocity variation along symmetry face

## **CONTOURS OF SURFACE NUSSELT NUMBER**

### **FIGURE 5.11**

The figure shows contours of surface Nusselt number along chip walls, velocity inlet and pressure outlet. At velocity inlet surface Nusselt number is found out to be zero shown by blue color as there was no convection. The surface Nusselt number is maximum at the front of first chip showing maximum convection heat transfer and slowly decreases as we move from first chip to last chip. At Pressure outlet surface Nusselt number is found out to be zero shown by blue color as there was no convection. Along the edges of velocity inlet face we have convection heat transfer from isothermal wall to cooling air shown by green color in a thin region near the inlet wall edges.

### **FIGURE 5.12**

The figure shows contours of surface Nusselt number along isothermal walls. Near to the velocity inlet face, the isothermal walls have maximum convection heat transfer as there is maximum temperature difference between inlet air and isothermal wall. This is shown by change of color from red at velocity inlet face to blue at pressure outlet face.

### **GRAPHS OF SURFACE NUSSELT NUMBER**

#### **FIGURE 5.13**

The graph shows variation of surface Nusselt number along symmetry. For chip walls the variation decreases from maximum to minimum along all the three chips. At the tip of first chip, the surface Nusselt number is maximum which slowly decreases for other two chips as shown in graph. In the space between the chips, the surface Nusselt number is found out to be zero. Along the top horizontal edge of isothermal number, the surface Nusselt number continuously decreases without any discontinuity.

#### **FIGURE 5.14**

The graph shows variation of surface Nusselt number along plane XY. For chip walls the variation decreases from maximum to minimum along all the three chips. At the tip of first chip, the surface Nusselt number is maximum which slowly decreases for other two chips as shown in graph. In the space between the chips, the surface Nusselt number is found out to be zero. Along the top horizontal edge of isothermal number, the surface Nusselt number continuously decreases without any discontinuity.

## RESULTS AND CONCLUSION

A more than hundred simulations have been done in order to grid independence check. The simulation has been simulated by meshing the volume of horizontal channel by taking different mesh elements and their type and different interval counts and interval size. The values found for Surface Nusselt Number are approximately same for last ten simulations. Hence grid independence has been checked. The average value for surface Nusselt number for the whole volume of the horizontal grid was found out to be 214. The value is found maximum along symmetry and is 217.8 in plane-XY

The Reynolds's Number is calculated to be 4792.11 which is for turbulent flow in rectangular duct. Hence our initial assumption of inlet velocity 2m/s and turbulent model is found to be correct.

These are the final report of fluxes being calculated

### MASS FLOW RATE

zone 6 (heatfluxwall): 0

zone 7 (isothermalwall): 0

zone 3 (prout): -0.0030620957

zone 5 (velin): 0.0030625092

net mass-flow: 4.1350722e-07

### TOTAL HEAT TRANSFER RATE

zone 6 (heatfluxwall): 4.8000083

zone 7 (isothermalwall): 19.825752

zone 3 (prout): -30.038778

zone 5 (velin): 5.4586892

net heat-transfer: 0.045671463

### RADIATION HEAT TRANSFER

zone 6 (heatfluxwall): 0

zone 7 (isothermalwall): 0

zone 3 (prout): 0

zone 5 (velin): 0

net rad-heat-transfer

## APPLICATION OF CFD

CFD is useful in a wide variety of applications and here we note a few to give you an idea of *its* use in industry. The simulations shown *below* have been performed using the FLUENT software. CFD can be used to simulate the flow over a vehicle. For instance, it can be used to study the interaction of propellers or rotors with the aircraft fuselage. The following figure shows the prediction of the pressure field induced by the interaction of the rotor with a helicopter fuselage in forward flight. Rotors and propellers can be represented with models of varying complexity. The temperature distribution obtained from a CFD analysis of a mixing manifold is showing below. This mixing manifold is part of the passenger cabin ventilation system on the Boeing 767. The CFD analysis showed the effectiveness of a simpler manifold design 'without the need for field testing.

Bio-medical engineering is a rapidly growing field and uses CFD to study the circulatory and respiratory systems. The following figure shows pressure contours and a cutaway view that reveals velocity vectors in a blood pump that assumes the role of heart in open-heart surgery. CFD is attractive to industry since It is more cost-effective than physical testing. However, one must note that complex flow simulations are challenging and error-prone and it takes a lot of engineering expertise to obtain validated solutions

Snap was selected under option. Lines were selected to the right of the grid. The grid options were control-right clicked.

## BIBLIOGRAPHY

- [1]. Bergles A.E and Bar Cohen, A, Immersion cooling digital computers, cooling of electronic systems, ;Kakae, S, Yanu, H, and Hijikata, K, Khaver Academic publishers, Baston M.A pg-539-621, 1994.
- [2]. [www.fluent.com/ software/ gambit](http://www.fluent.com/software/gambit).
- [3]. [www.hlrn.de/doc/ fluent](http://www.hlrn.de/doc/fluent)
- [4]. Incropera FP, convection heat transfer in electronic equipment cooling, Journal of heat transfer 110 (1988).
- [5]. Kennedy KJ, Zebib A, combined free and forced convection between vertical and parallel plates, some case studies. International journal of Heat and Mass Transfer 26 (1990).
- [6]. Danielson RD, tousignant, L, and Bar-Cohen, A saturated pool boiling characteristics of commercially available per flourinated liquids, proc of ASME/ ISME Thermal engineering joint conference.
- [7]. Chrsler, G.M, chu, r.C. and Simons, RE jet impingement Boiling of a dielectric coolant in narrow gaps, IEEE trans. CHMT -part A, VOL18(3),pg 527-533,1955
- [8]. 8086 Microprocessors by D.V. Hall.
- [9]. 8085 Microprocessors by Gaonkar.
- [10]. Computational Methods for Fluid Dynamics by J.H.Ferziger& M.Peric (3<sup>rd</sup> edition)
- [11]. Computational fluid Dynamics by John D. Anderson.
- [12]. Numerical Heat Transfer and Fluid Flow by Suhas V. Patankar